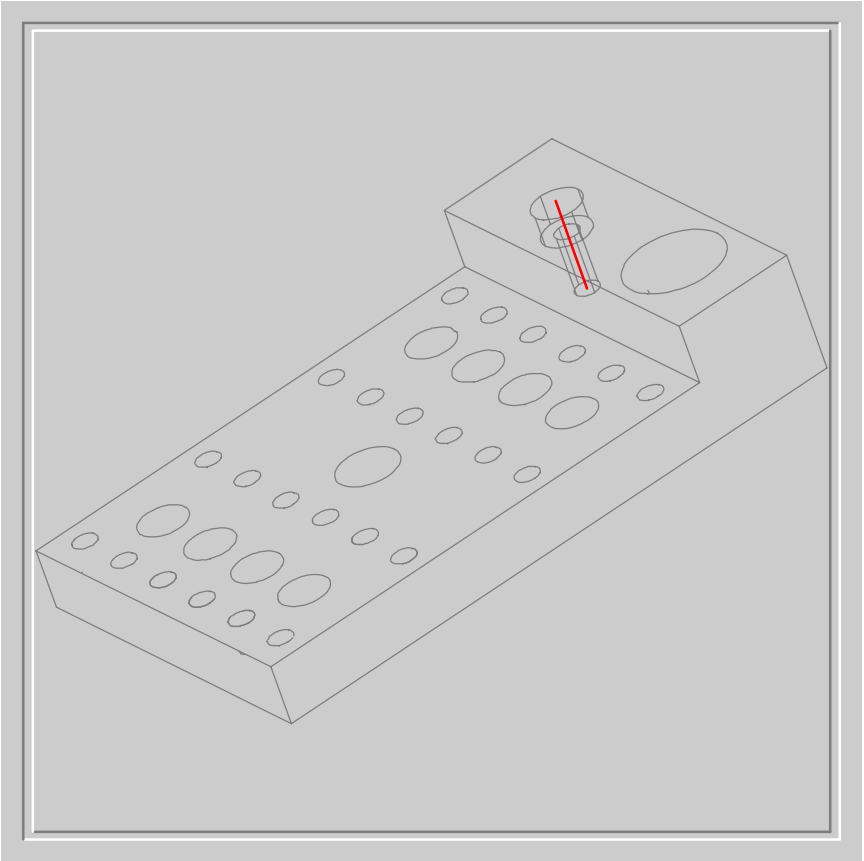


Setting Machining Parameters for Hole–Making Operations

I-DEAS® Tutorials: Milling Projects and Turning Projects

In this tutorial, you'll learn how to specify the options and parameters for cutting holes. You'll learn how to define the type of hole to cut and its depth by creating different hole-making operations.

Learn how to:

- create a countersink operation
- create a drill operation
- create a tap operation
- define the point depth

Before you begin...

Prerequisite tutorials:

- all tutorials under the Modeling Fundamentals menu
- Introduction to Generative Machining
- Building a Setup Assembly
- Generating In-process Stock and Checking Validity
- Working with Tools and Tool Catalogs
- Picking Holes

Retrieve the model file that you created in the Picking Holes tutorial.

File
Open

Model File Name: holes
OK

Make sure you're in the following application and task:

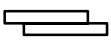
Application: Manufacturing
Task: Generative Machining

Set your units to millimeters.

Options,
Units

mm (milli newton)

Recovery Point



File
Save

Warning!
If you're prompted by I-DEAS to save your model file, respond:

No

Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

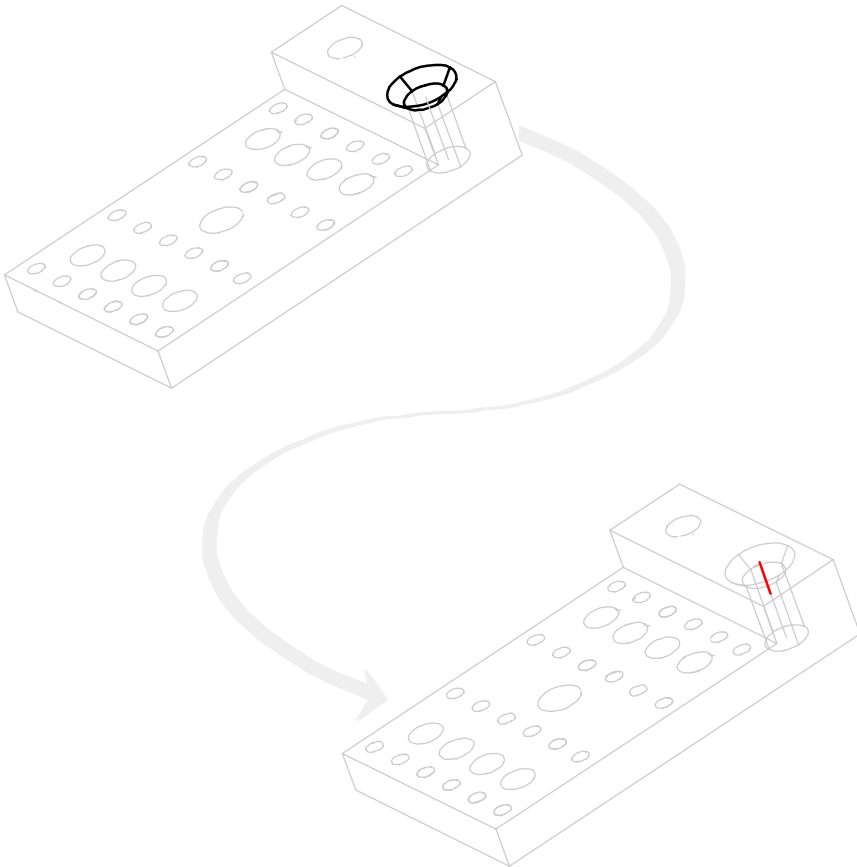
If you make a mistake at any time between saves and cannot recover, reopen your model file to the last save and start over from that point.

Hint
To reopen your model file to the previous save, press Control-Z.

Create a countersink operation 1 of 6

In the next steps, you'll create a countersink operation to machine a chamfer on the part. You can define the cycle depth by either the diameter or the distance of the chamfer. The diameter is measured at the top of the chamfer, while the distance specifies the depth of the corner break.

The software calculates the depth needed to produce the chamfer based on the cutting tool's point angle.



What: Create an opgroup and a countersink operation.

How:

NC Job Planning form

Deselect *Drill 40mm Dia Holes* by pressing the Control key and selecting *Drill 40mm Dia Holes*.

OpGroup Specification form

Name: Machine Thru Tapped Hole

Operation Selection form

Category: Hole Making

Type: Countersink

Create

Don’t close the Operation Specification form.

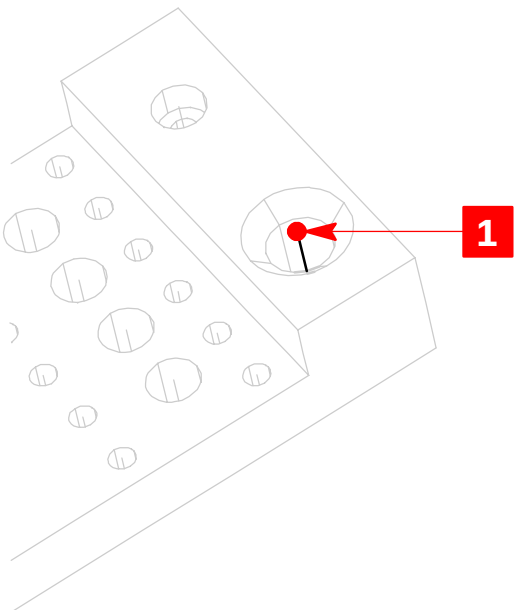
What: Pick the hole to be machined. The uppermost point represents the chamfer.

How:

Operation Specification form

Name: Machine Countersink

1 CP116



Hole Machining Order form

OK

Don’t close the Operation Specification form.

What: Create a countersink drill.

How:

Operation Specification form

Cutting Tool Specification—Drill form

Identifier: 80mm Countersink

Press the Tab key to move to the next field.

I-DEAS Warning

OK

Holder Diameter: 70

Shank Diameter: 25

Holder To Tip Dist: 75

Cutter Diameter: 80

Point Angle: 90

OK

Don’t close the Operation Specification form.

What: Enter the diameter at the top of the chamfer, 80mm.

How:

Operation Specification form

Machining Parameters: Cut form

 *Chamfer* (turn on)

 *Chamfer Size:* 80

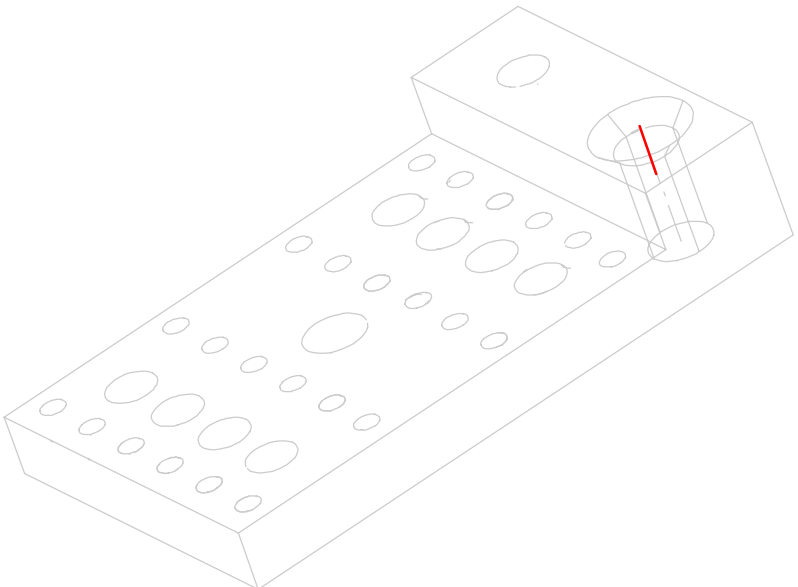
 *OK*

 Don’t close the Operation Specification form.

What: Generate the toolpath.

How:

Operation Specification form



Things to notice

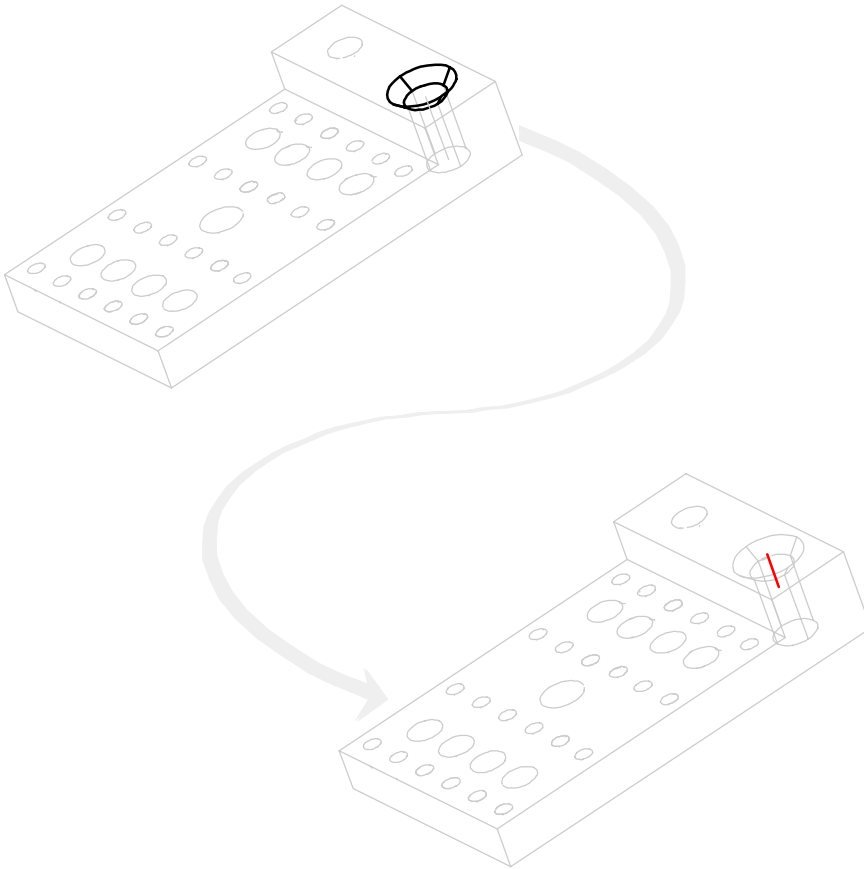
The generated toolpath machines the chamfer only and doesn't cut to the bottom of the hole.

Recovery Point

File
Save

Create a drill operation 1 of 5

For the next operation, you'll derive the cycle depth from the geometry itself, instead of entering the depth. You pick the hole, and the software calculates the cycle depth automatically. You'll also enter a distance beyond this depth for the full diameter to feed past the bottom of the hole.



What: Create a drill operation.

How:

NC Job Planning form

Machine Thru Tapped Hole

OpGroup Specification form

Deselect *Machine Counter Sink* by pressing the Control key and selecting *Machine Counter Sink*.

Operation Selection form

Category: Hole Making

Type: Drill

Create

Don’t close the Operation Specification form.

What: Name the operation and create a 50mm standard drill.

How:

Operation Specification form

Name: Drill thru Hole

Cutting Tool Specification—Drill form

Identifier: 50mm Standard Drill

Press the Tab key to move to the next field.

I-DEAS Warning

OK

Description: Standard drill

Holder Diameter: 75

Shank Diameter: 50

Holder To Tip Dist: 150

Max Depth of Cut: 125

Cutter Diameter: 50

OK

Don’t close the Operation Specification form.

What: Derive the cut depths from the geometry. You didn't have to pick the hole because the chamfer is still selected from the previous operation. The depth of the chamfer is 15mm. Then enter 5mm for the tool to feed past the bottom of the hole to ensure all the material is cleared away.

How:

Operation Specification form

Machining Parameters: Cut form

 *Get Hole Depths From Geometry*

 *Thru Clearance Distance*: 5

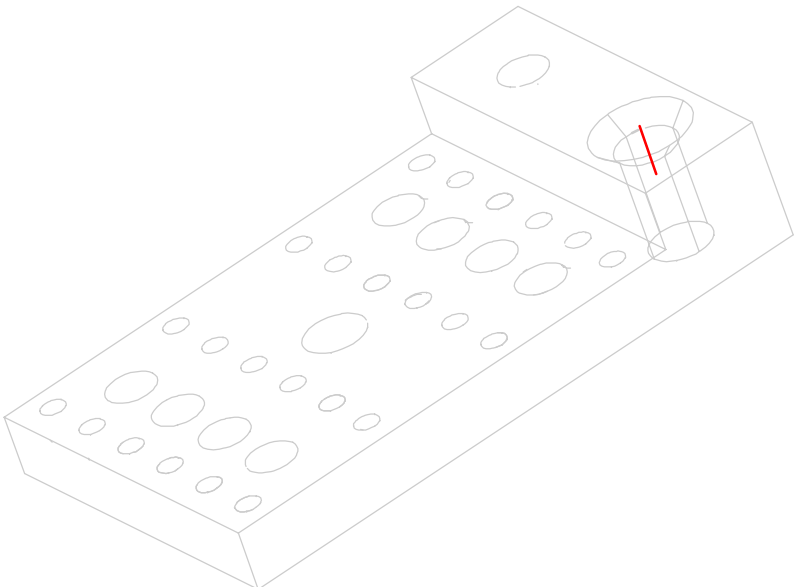
 *OK*

 Don't close the Operation Specification form.

What: Generate the toolpath.

How:

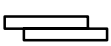
Operation Specification form



Things to notice

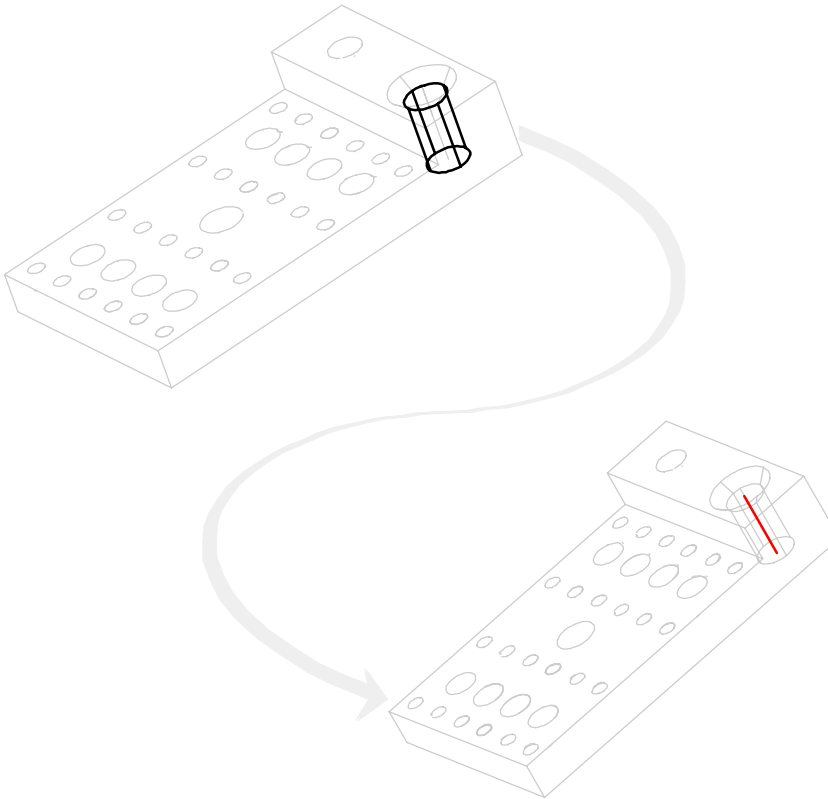
The drill operation cuts to the depth of the chamfer and then extends the cycle by the 5mm that you entered. Again, the tool doesn't cut to the bottom of the hole because the chamfer is selected.

Recovery Point



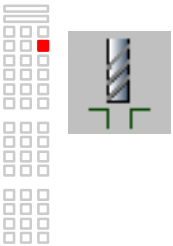
File
Save

In the next steps, you'll create a tap operation. You'll enter the hole depth—100mm—to calculate the cycle. You'll also learn how to define feedrate parameters for tapping operations.



What: Create a tap operation.

How:



Operation Selection form

Category: Hole Making

Type: Tap

Create

Don’t close the Operation Specification form.

What: Pick the area of the hole to machine.

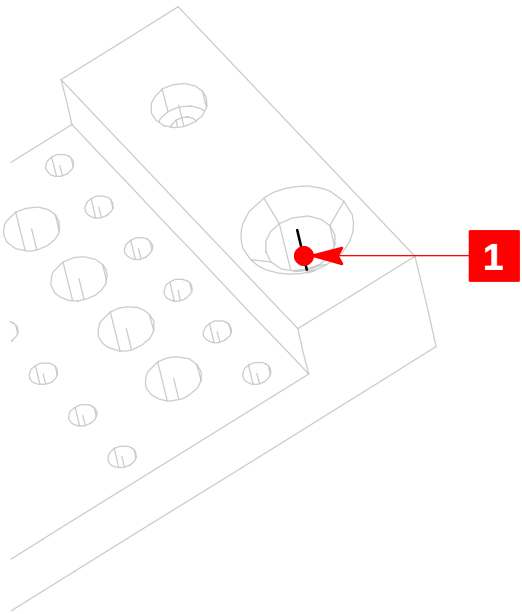
How:

Operation Specification form

Name: Tap Hole

Hole Machining Order form

1 CP114



OK

Don’t close the Operation Specification form.

What: Create a drill with a 41.5mm tap.

How:

Operation Specification form

Cutting Tool Specification—Drill form

Identifier: 41.5mm tap

Press the Tab key to move to the next field.

I-DEAS Warning

OK

Holder Diameter: 80
Shank Diameter: 25
Holder to Tip Dist: 200
Max Depth of Cut: 125
Cutter Diameter: 41.5
Other Attributes...

Pitch: 3
Dismiss

OK

Don’t close the Operation Specification form.

What: Enter the hole depth. When you enter the hole depth, the software calculates the toolpath so that the full diameter of the tool cuts to that depth. However, the full diameter hole depth excludes the length of the tool point. In this instance the hole depth is 85mm, so you'll be adding 15mm to the cycle depth.

Then specify the tapping override percentage at 96%. *Tapping Override %* specifies a percentage used for increasing or decreasing the feedrate when using tension/compression holders. The feedrate is calculated from the spindle speed and lead or pitch. *Tapping Override %* is multiplied by this feedrate.

How:

Operation Specification form

Machining Parameters: Cut form

Hole Depth: 100

Cut...

Feeds & Speeds...

Spindle Speed: 50

Tapping Override%—On

Tapping Override%: 96

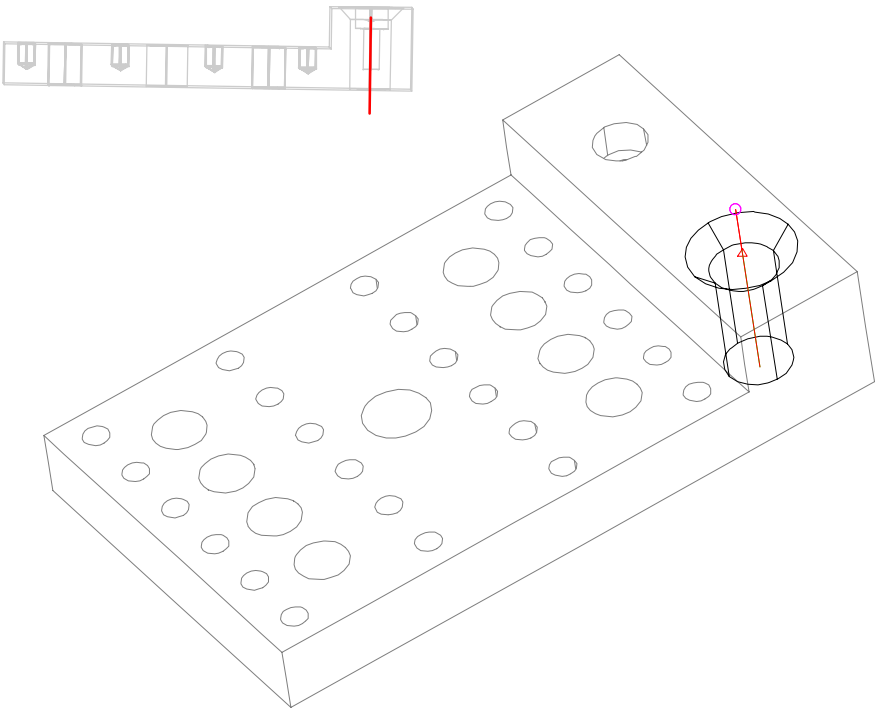
OK

Don't close the Operation Specification form.

What: Generate the toolpath.

How:

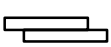
Operation Specification form



Things to notice

If you animate the tool, you'll notice that the full diameter cuts 15mm beyond the bottom of the hole. Keep in mind that when you enter the hole depth, it doesn't account for the point of the tool, only its full diameter. Notice how the tool point cuts past the bottom edge of the hole also.

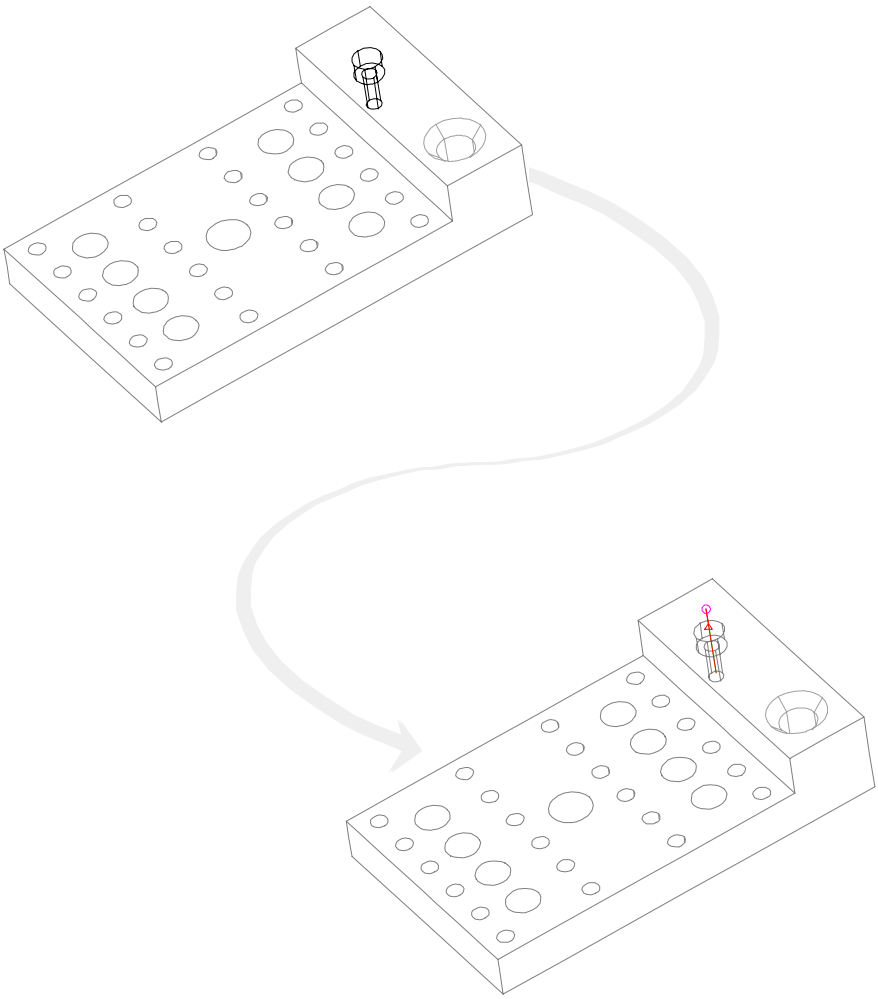
Recovery Point



File
Save

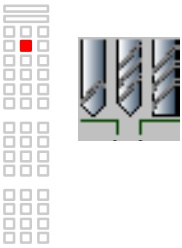
Define the point depth

In these last steps, you'll create a drill operation to machine another hole. You'll learn how to measure the depth of a hole and how to drill the hole based on the point of the tool.



What: Create a new opgroup.

How:



OpGroup Specification form

Name: Machine Counterbored Hole

Operation Selection form

Category: Hole Making

Type: Drill

Create

Don’t close the Operation Specification form.

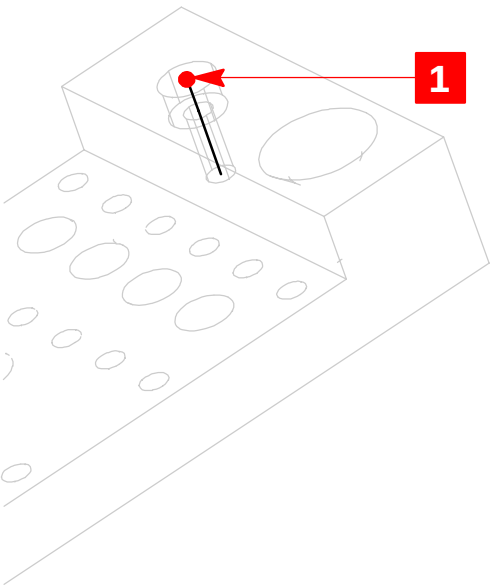
What: Pick the top of the hole to be machined.

How:

Operation Specification form


Name: Drill Cbore

1 CP118



Hole Machining Order form


OK

Don’t close the Operation Specification form.

What: Create a 40–20 Subland drill.

How:

Operation Specification form

Cutting Tool Specification—Drill form

Identifier: 40–20 Subland Drill

Press the Tab key to move to the next field.

I-DEAS Warning

OK

Description: 40mm drill diameter 20mm pilot diameter x 50mm pilot length

Cutter Diameter: 40

Style: Double Diameter

Minor Diameter: 20

Minor Depth: 50

Transition Angle: 180

OK

Don’t close the Operation Specification form.

What: Define the point depth by measuring the distance of the endpoints.

How:

Operation Specification form

Machining Parameters: Cut form

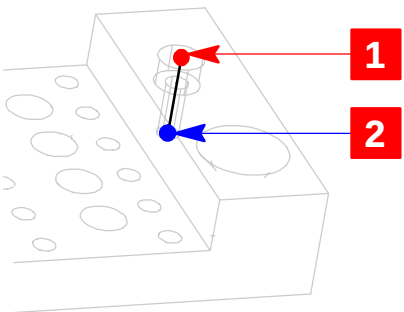
Point Depth

Measure (next to *Point Depth*)

1 CP118

2 CP119

(twice)



Read the I-DEAS List to ensure the depth equals 75mm.

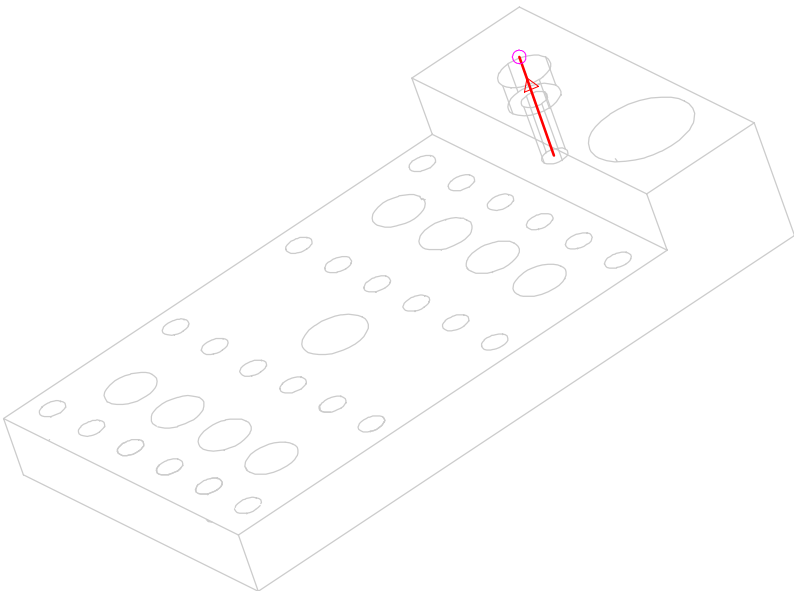
OK

Don’t close the Operation Specification form.

What: Generate the toolpath.

How:

Operation Specification form



Things to notice

The software calculates the cutting motion until the tip of the drill contacts the bottom of the hole. Notice how the point of the tool doesn't extend beyond the hole for point depth (unless you enter a distance beyond the hole).

Recovery Point

File
Save

Tutorial wrap-up

You’ve completed the Setting Machining Parameters for Hole-Making Operations tutorial.